OpenFOAM Simulations for MAV Applications

Syed Zahid*, A. Rajesh§, M.B. Subrahmanya†, B.N. Rajani†

*Student, Dept. of Mech. Engg, SDM, Dharwad, 580 002, India
§Project Assistant, CTFD Division, CSIR-NAL, Bangalore 560 017, India
†Scientist, CTFD Division, CSIR-NAL, Bangalore 560 017, India

Abstract

In the present study open-source CFD tool OpenFOAM has been used to simulate (i) two dimensional turbulent flow past airfoils and (ii) three dimensional turbulent flow past rectangular thin wing and fuselage used for MAV (Micro Air Vehicle) applications. The results obtained from the present simulation have been compared with NAL incompressible flow solution code 3D-PURLES and the available measurement data. The turbulent flow simulations have been carried out by solving the Reynolds Averaged Navier Stokes (RANS) equations.

Keywords: OpenFOAM, MAV, NACA0012, SD7003, fuselage.

1. Introduction

The present work primarily concentrates to study the capability of the OpenFOAM CFD tool to predict the aerodynamic and flow characteristics of the MAV wings which operate at relatively low Reynolds number. OpenFOAM is free software with the full source code available. This facility enables the user to modify the code as per their application and add new capabilities making it a versatile tool for researchers. The researchers using this tool have added advantage as the tool gets good and fast community support in various fields of application. There is a continuous updation and development of this tool by various groups working across the world. The versatility and no cost make this tool very popular among the students and research community. Recently it is being widely used to solve industrial problems and for defense applications. In this work the capability of OpenFOAM is being established with the available in-house structured flow code 3D-PURLES (Three Dimensional - Pressure based Rans LES solver) which has been extensively validated and used for various research projects sponsored by DRDO, AR&DB, NRB etc.

Results and Discussion

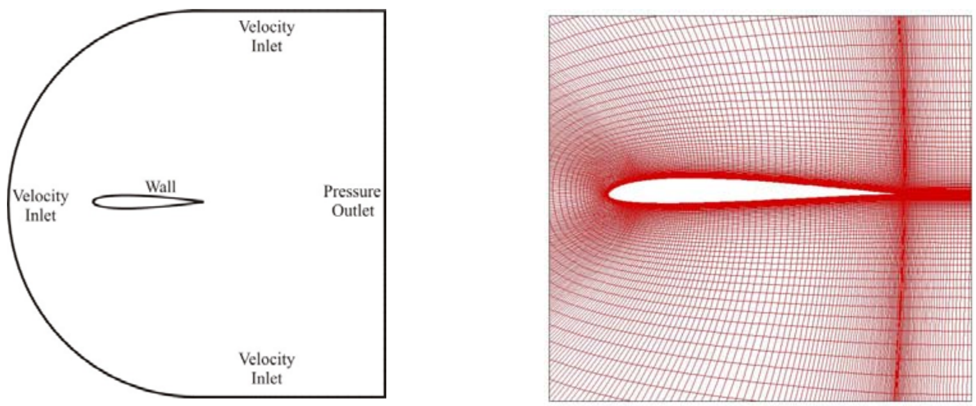
1.1 Turbulent Flow past Airfoils

The section discusses the results obtained for flow past NACA0012 airfoil and SD7003 low Re airfoil. The grid size and boundary condition used for both the computations are identical and is shown in Fig. 1. This simulation uses simpleFoam (steady state solver for turbulent flow) in which the gradients are discretised using the Gauss linear scheme and the convective terms are discretised using Gauss linear upwind. The convergence criterion was set to 10^{-5} and the eddy viscosity at far field is taken to be 10 times to the laminar viscosity for all computations.

The turbulent simulations for NACA0012 airfoil at $Re=1\times10^6$ have been carried out using the SA turbulence model. The comparison of the variation of aerodynamic coefficients for NACA0012 airfoil obtained is shown in Fig. 2. The variation of lift coefficient is observed to follow the expected trend and matches well with the 3D-PURLES [1] results and measurement data [2] up to the stall angle ($\alpha \leq 12^0$). Both the computations have predicted a delayed stall when compared to the measurement. The maximum lift coefficient predicted by OpenFOAM and 3D-PURLES are higher than that of the measurement data with OpenFOAM having the highest lift coefficient. The drag coefficient (Fig. 2(b)) obtained from both the codes are almost identical but is slightly over predicted when compared to measurement. The SD7003 airfoil simulations at $Re=6\times10^4$ have been carried

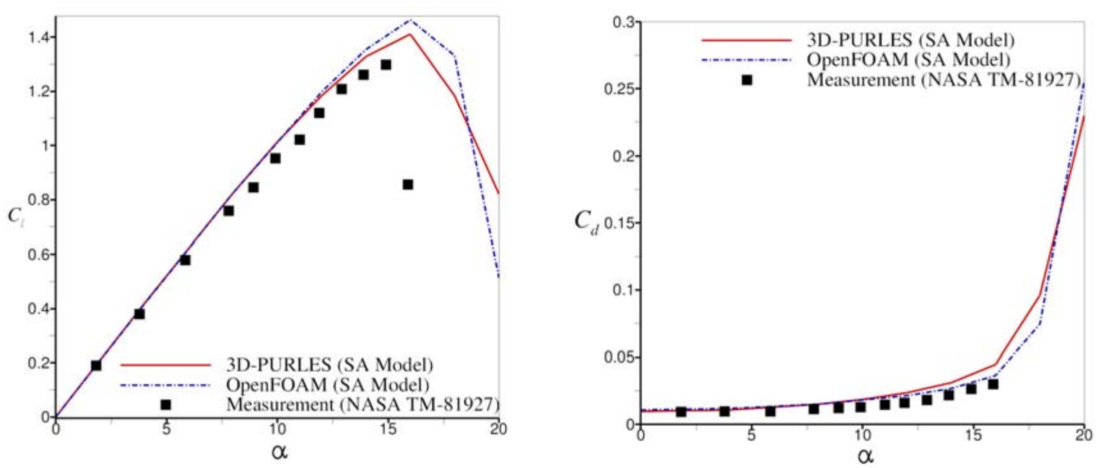
*Corresponding author. Email: s.zahid4u@gmail.com

out using SST turbulence model because SA turbulence model had some convergence issues at higher angle of attack. The variation of aerodynamic coefficients obtained for SD7003 airfoil are shown in Fig.3. The OpenFOAM and 3D-PURLES [3] results for lift coefficient (Fig. 3(a)) are observed to be in close agreement up to the stall angle and matches reasonably well with the measurement data [4]. Both the codes have predicted the same stall angle ($\alpha = 11^0$) which is slightly earlier compared to measurement ($\alpha = 13^0$). However the maximum lift obtained by 3D-PURLES is over-predicted compared to OpenFOAM and measurement. Similar to the NACA0012 airfoil, the drag coefficients predicted by both the codes are in close agreement with each other.



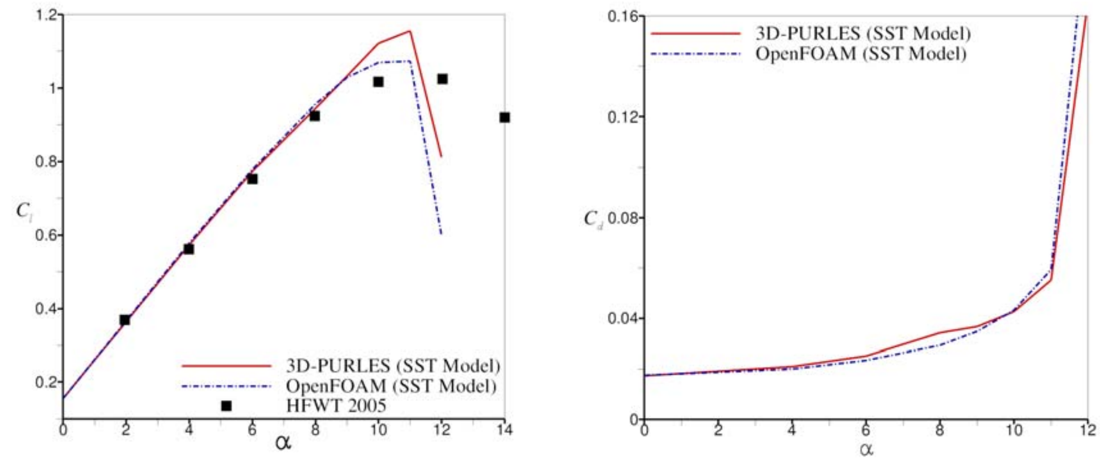
(a) Boundary conditions (b) Grid for SD7003 (521×101, y^+<1)

Figure 1: Boundary condition and grid used for flow past airfoils



(a) Lift coefficient (b) Drag coefficient

Figure 2: Variation of aerodynamic coefficients for flow past NACA0012 airfoil (Re = 1.0x10^6, SA)

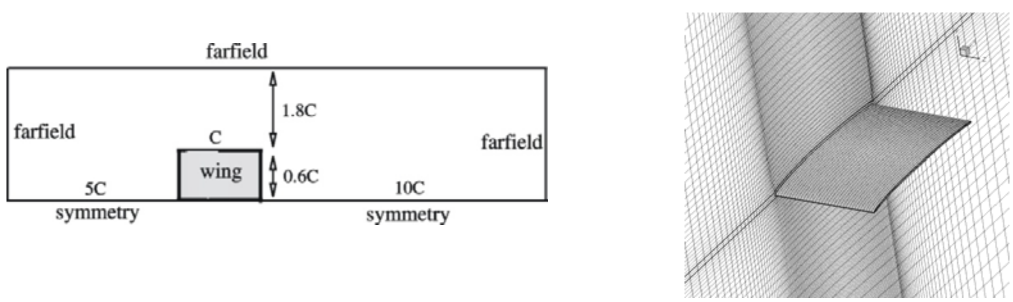


(a) Lift coefficient (b) Drag coefficient

Figure 3: Variation of aerodynamic coefficients for flow past SD7003 low Re airfoil (Re-60,000, SST)

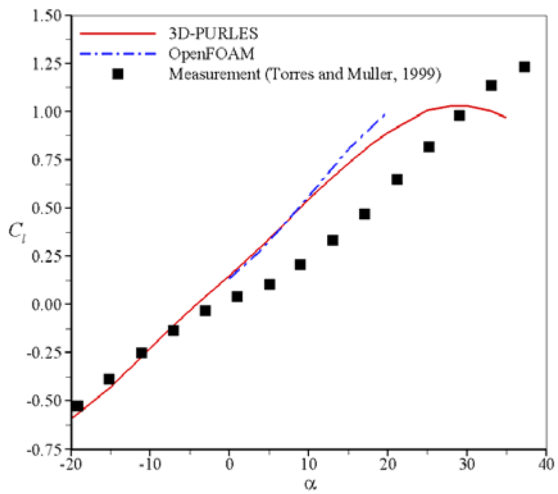
2.2 Flow past a Rectangular thin wing

The simulations for the rectangular wing having a cross section of selig4083 camber line with one percent thickness and semi span 0.6C have been carried out at $Re = 2.4\times10^5$ based on the root chord length (*C*) and a wind speed of 14 m/s. These simulations have been carried out only for few angles of attack (0 to 20). The grid (2.77 million control volumes) for this simulation has been generated using the ICEM CFD grid generation tool. The computational domain and the boundary conditions used for the present simulation are shown in Fig. 4. The gradient and convective terms in the simpleFoam solver are discretised using the same scheme as used in the airfoil simulations. For the present computations the convergence criterion was set to 10^{-4}. The computation was carried out using standard SA model. The variation of aerodynamic coefficients obtained from the present simulation have been compared with 3D-PURLES results [5] and experimental values of Torres and Mueller [6] available for similar rectangular wing at slightly different flow conditions. The aerodynamic coefficients obtained from this simulation are in good agreement with the 3D-PURLES. The lift coefficient curve indicates that the lift obtained by OpenFOAM is slightly higher when compared to 3D-PURLES at α = 20. The non-linearity observed in the measurement at lower positive angles of attack could not be captured in both the simulations because of the fully turbulent assumption.

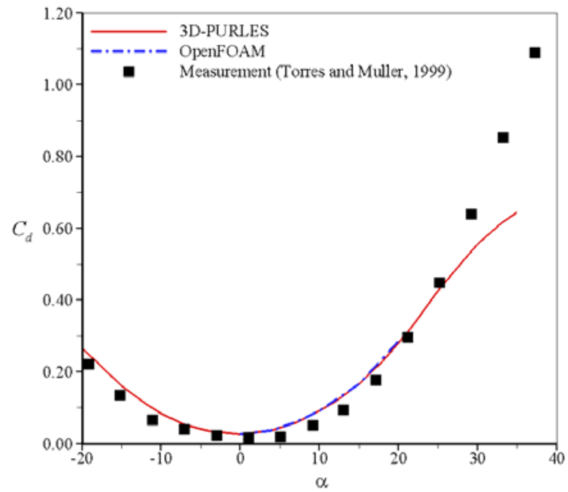


(a) Boundary conditions (b) Surface grid on symmetry plane and wing

Figure 4: Grid and boundary conditions for flow past rectangular thin wing



(a) Lift coefficient

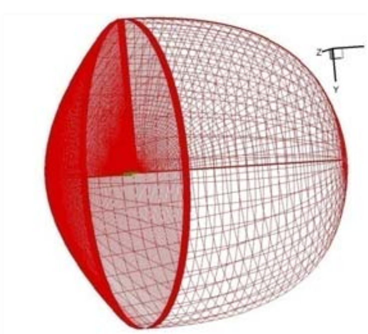


(b) Drag coefficient

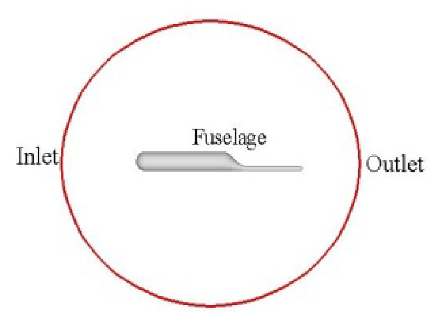
Figure 5: Variation of aerodynamic coefficients for flow past rectangular wing (Re=2.4×10^6, SA)

2.3 Flow past Slybird Fuselage

OpenFOAM has been used to simulate flow past Slybird-V2 fuselage at Re=1.4×10^6 (based on the fuselage length and a wind speed of 15m/sec) at α=12^0. The aerodynamic coefficients obtained are compared with NAL measurement data and 3D-PURLES computational data. The simulation uses a structured grid 54 x 82 x 41 generated using POINTWISE (Fig 5.12) which is same as that used in 3D-PURLES simulations. The turbulent simulations have been carried out using simpleFoam having the same input as discussed in section 2.2 with standard k-ε model. OpenFOAM (0.0128) and 3D-PURLES (0.0147) have under predicted the lift coefficient compared to measurement data (0.0216). However, the drag obtained from OpenFOAM (0.0087) is grossly under predicted when compared to 3D-PURLES (0.0144) and measurement (0.0137). This difference may be attributed to the coarse grid size resolution as OpenFOAM is quite sensitive to the grid quality. Refining the grid size and quality may improve the drag coefficient value.

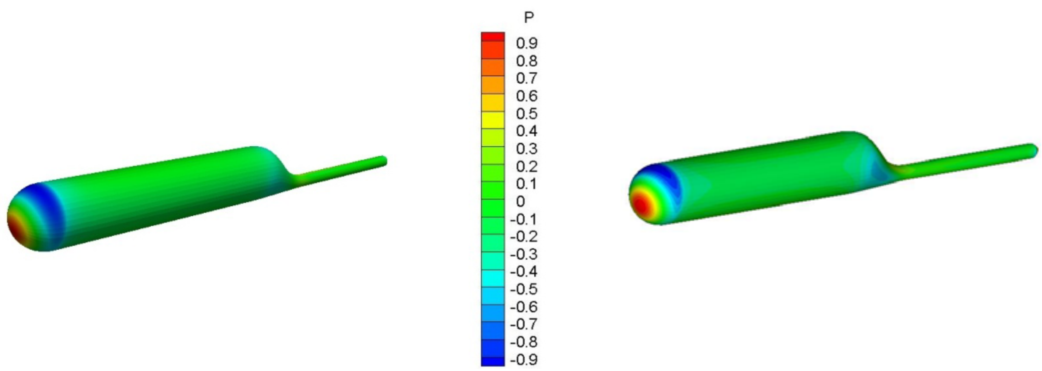


(a) Grid (54 × 82 × 41, y+ ~ 30)



(b) Boundary conditions

Figure 6: Grid and boundary conditions for flow past fuselage



OpenFOAM 3D-PURLES

Figure 7: Pressure contours at $\alpha = 12^0$ ($k-\varepsilon$ model, Re=1.4 x 10^6)

2. Conclusion

The capability of OpenFOAM has been established to simulate flow past MAV configuration. The aerodynamic coefficients predicted using OpenFOAM for all the cases are in reasonable agreement with the NAL in-house flow solution code 3D-PURLES and measurement data.

Aknowledgements

The authors wish to express their thanks to the Director and head CTFD Division, CSIR-NAL, Bangalore for permitting us to publish this paper. We also deeply thank Mr. D. S. Kulkarni, Scientist for his valuable technical discussions and suggestions.

References

[1] T. Venkatesh, D. S. Kulkarni and B. N. Rajani, CFD Simulation Using FLUENT and RANS3D - A Validation Exercise. CSIR-NAL Project Document, *PD CF 1203*, May 2012.

[2] D.H. Charles, Two Dimensional Aerodynamic Characteristics of the NACA0012 Aerofoil in the Langely 8-foot Transonic pressure tunnel, *NASA TM* 81927. 1981.

[3] A. Rajesh, M. B. Subrahmanya, D. S. Kulkarni and B. N. Rajani, Flow past low Re airfoils. *NPMICAV interim project review report,* Jan 2014.

[4] M.V. Ol, B.R. McAuliffe, E.S. Hanff, U. Scholz, and C. Kalher., Comparison of Laminar separation bubble measurements on a low Reynolds number aerofoil in three facilities. *American Institute of Aeronautics and Astronautics paper, 2005-5149*, 2005.

[5] Pradeep Shetty, M. B. Subrahmanya, D. S. Kulkarni and B. N. Rajani, CFD Simulation of Flow Past Micro Air Vehicle Wings, *International Journal of Aerospace Innovations*, vol 5(1), pp 19-27, 2013

[6] G. E. Torres and T. J. Mueller., Aerodynamic Characteristics of Low aspect ratio wings at low Reynolds number. In T. J. Mueller, editor, *Fixed and Flapping wing aerodynamics for Micro Air Vehicle applications*, volume 195 of *Progress in astronautics and Aeronautics*, AIAA pp 115–141, 1999.